International Journal of Mechanical and Production
Engineering Research and Development (IJMPERD)
ISSN(P): 2249–6890; ISSN(E): 2249–8001
Vol. 10, Issue 1, Feb 2020, 59–72

EFFECT OF DIFFERENT PORT ANGLE IN AN IC ENGINE ON THE SWIRL AND MASS FLOW RATE

ADHIRATH MANDAL[1] & HAENGMUK CHO[2]

[1]*Doctorate Student, Department of Mechanical Engineering, Kongju National University, Republic of Korea*

[2]*Professor, Department of Mechanical Engineering, Kongju National University, Republic of Korea*

ABSTRACT

Diesel Engine is very popular and is being used in many industries. It is used in automobile for transmitting power to the wheels, used in generator sets for producing electricity and also used in the constructing machinery to mix the construction materials. Diesel engine is also used in telecommunication infrastructure and mining facilities. Internal combustion engine has four strokes. Intake, compression, expansion and exhaust strokes are the common four strokes in an internal combustion engine. Intake and compression stroke are the important strokes out of the four strokes in an IC engine. These two strokes define the air flow pattern in the combustion chamber, which then, render the fuel injection condition during compression stroke. However there are some problems such as decreased efficiency and enhanced emission are due to improper diffusion. Emission characteristics and combustion performance are directly affected by the air fuel mixing. Research communities are very interested in the internal combustion engine, air flow characteristics. To improve the efficiency of the engine and to reduce the emissions, research of basic flow dynamics in the cylinder is essential. This paper shows simulation analysis of different angles of the valve. The valve had different angles of 30^0, 45^0 and 60^0. Swirl and mass flow rates are compared in this paper.

KEYWORDS: IC Engine, Port Flow Simulation, Swirl, Mass Flow Rate & Valve

Received: Oct 01, 2019; **Accepted**: Oct 21, 2019; **Published**: Nov 30, 2019; **Paper Id.**: IJMPERDFEB20206

Original Article

1. INTRODUCTION

Diesel Engine is very popular and is being used in many industries. They are used in automobile for transmitting power to the wheels, used in generator sets from producing electricity and also used in the constructing machinery to mix the construction materials. Diesel engine is also used in telecommunication infrastructure and mining facilities. In many situations, the generator sets are used in remote locations where electricity is not available. To run the diesel generator sets, we need diesel or a hybrid fuel with renewable energy. Diesel engine is very commonly used because of its good fuel efficiency, cheap price of fuel and good reliability. Internal combustion engine has four strokes. Intake, compression, expansion and exhaust strokes are the common four strokes in an internal combustion engine. Intake and compression stroke are the important strokes out of the four strokes in an IC engine. These two strokes define the air flow pattern in the combustion chamber which then, render the fuel injection condition during compression stroke. However, there are some problems such as decreased efficiency and enhanced emission are because of improper diffusion. Emission characteristics and combustion performance are directly affected by the air fuel mixing. The emission performance could be improved by circulating air uniformly and widely in the combustion chamber 13.

But in recent years, we have seen that the price of diesel fuel is rising, as the fossil fuels are depleting and global warming issue is also rising. In many researches, biodiesel and biodiesel blends have been considered as a

good alternative to diesel. But with the use of biodiesel, NO_x emission increases. The increase in the NO_x emission is because of the high percentage of oxygen during combustion 4.

Because of the above reasons, the research community is very much interested in the internal combustion engine, air flow characteristics. To improve the efficiency of the engine and to reduce the emissions, research of basic flow dynamics in the cylinder is essential. The inlet ports in an IC engine should be designed in such a way that the swirl and tumble in the cylinder could be enhanced. Swirl and tumble are illustrated in figure 1. The yellow arrow refers to the swirl in the cylinder and the blue arrow refers to the tumble in the cylinder. As the air, which enters from the inlet port, the energy of the inlet jet will retain in the cylinder through the combustion cycle, which would increase the efficiency by mixing the fuel and air better. The inlet port in a compression ignition engine are so designed in such a way that the swirl is more present in the cylinder, but in the case of gasoline engine, the tumble is more present in the cylinder.

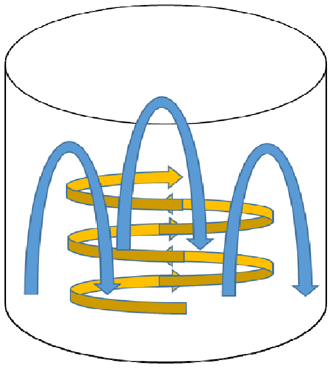


Figure 1: Description of Swirl and Tumble.

Swirl and tumble are directly linked with combustion efficiency. But too much swirl would increase the soot emission and consumption of fuel which is due to the fuel spray collision 4. In many researches, it was found that the inlet manifold was modified to increase the swirl and was validated with experimental data. It was found that the experimental data and simulation data were similar.

The main objective of this paper is to compare swirl and mass flow rate inside the combustion chamber for varying angle of the port with the help of CFD port flow simulation.

2. LITERATURE REVIEW

Mohammad Abdullah Mohamad Johar et al. researched the relationship between the geometry of the intake port and the mass flow rate in the cylinder head. The amount of air-fuel mixture moving through the intake port per second is referred as the mass flow rate. Ansys workbench simulation was utilised to study the mass flow through the intake port. As the cylinder and the intake port are symmetrical, the simulation was conducted only on the half part. The air-fuel mixture pressure was kept constant throughout the intake port during simulation. The valve opening was between 4mm to 10mm, same in the case of super flow. With these boundary conditions, the mass flow rate was recorded which corresponds with the valve lift. Intake port geometry was modified for this study. All the intake ports were used for the simulation and the mass flow rate varied with different intake port. It was noticed that the mass flow rate could be improved by 7.1% at 8mm lift of the valve. It was concluded that the highest mass flow rate could be achieved when the air is attached to the surface. But perfect intake port is only possible theoretically, which has straight intake port. Mass flow rate could be improved by reducing the sharp turns of the intake port 6. David Rathnaraj Jebamani et al. researched about the effectiveness of helical

port and the tangential port. It was a known fact that, helical port was better to obtain maximum swirl when compared to the tangential port. In this condition, volumetric efficiency was least sacrificed. Design of the swirl was so designed in such a way that the swirl ration was reduced, as at higher speed, it would reduce the emission. But it was observed that the helical port increased the particulate smoke emission at lower speed and also the mixing time of the air and fuel. It is very necessary to have an ideal swirl ratio for reduction of emission and the proper combustion. Therefore, to enhance the combustion in the engine cylinder, it is important to have changing swirl. The charging quantity and the swirl can easily be controlled by opening of the valve. Steady flow rig experiment was used to study the swirl when a helical intake port was used at different operating conditions. W06DTIE2 engine was used with different valve openings to experiment and the variation in the swirl. The area of the inlet port is varied by sliding the swirl plate.

This study was based on the swirl optimized combustion system, by varying the swirl plate mechanism which was studied experimentally and was compared with the results of the CFD. CFD was studied on a direct injection diesel engine on the port-cylinder system. The study was conducted under steady flow rig. Results showed that the variable swirl increased for 70% opening of the intake port and the swirl was decreased for 50% opening. It was concluded that the high swirl could reduce the NO_x and particulates. These are the two main pollutants, and can be reduced by increasing the swirl. It was also highlighted that the CFD tool could be used for optimisation of the intake system and this tool will reduce the experiments to be carried out 8. Soonseong Hong et al. studied the engine intake manifold and implemented design for six sigma in the intake manifold to optimize it. The main aim of this study is to increase strength of swirl and also to maximize the mass flow rate in the cylinder chamber. Factors which affect the mass flow rate and the swirl strength are prime pipe section shape, plenum shape, port diameter, primary and secondary length etc. All these factors were adopted. To calculate the mass flow rate and the swirl strength, L18 orthogonal table was used for CFD simulation calculation. From the results, it was clear that with S/N ratio was increasing for both swirl and mass flow rate. Which stated that the with increasing speed the combustion performance improved. By increasing the beta value using DFSS process, optimum mass flow rate was obtained, under the PDA system 9. S. Bari et al. researched the usage of biodiesel as an alternative fuel, as it is renewable and behaves similar to diesel fuel in an engine. It was also noticed that the amount of carbon dioxide emission is cancelled by the plant creating biodiesel raw material. Though biodiesel have disadvantage of having heavier molecules and higher viscosity which makes the atomisation and mixing with air slow and therefore the efficiency of the combustion is low. To solve these problems, guide vane swirl and tumble device was set up to increase the turbulence in the combustion chamber to increase the efficiency of the combustion. With the use of curved vanes temperature and pressure inside the cylinder were higher as compare to without vanes. Curved vanes showed higher tumble and cross tumble. Curved vanes had lower TKE (Turbulence Kinetic Energy). Fuel penetration was reduced with the GVSTD initially, but before TDC, the injection length increased. Increased injection length resulted in more carbon deposit on the piston. With the GVSTD, the cone angle was narrow 11. Haiqin Zhou et al. proposed a separate swirl combustion system through which air efficiency would be improved. Analysis of combustion and combustion performance was carried out using 3D simulation. Sensitivity analysis, Soot emission and indicated power were carried out by chamber geometry. Chamber geometry was optimised by the results of the sensitivity analysis. Emission and combustion performance with the best chamber geometry of single swirl combustion chamber was compared with the double swirl combustion chamber with different speed in a single cylinder diesel engine. The simulation results showed that the fuel distribution in inner and outer chamber was affected because of angle of inner chamber, SSCS combustion performance was influenced greatly. Second important parameter for optimization was the depth of the separated chamber, then the outer chamber, and then the

separated chamber diameter. Lastly, the volume ratio of inner to outer chamber and also the angle of the outer chamber. Best geometry was determined after examining the different velocity distribution, fuel/air mixture, soot emission and combustion performance with different SSCS chamber geometries,. Combustion and emission performance of DSCS was compared with SSCS. It was found that the indicated thermal efficiency was higher and had lower break-specific fuel consumption, compared to DSCS at different engine speeds 12. Himanth Kumar H Y et al. studied the 3D model and conducted numerical analysis. Numerical analysis was conducted on a 46mm diameter of inlet port, 43mm diameter of valve and the cylinder having 562mm length and 93.65mm of diameter. The focus of the study was to check the effect of valve lift on the fluid flow in the cylinder. It was known that the velocity of the flow will change for different valve lifts. It was concluded that, for achieving good efficiency, and to achieve low emission, it is very important to study the flow inside the intake port. With the help of CFD tool, the flow characteristics were studied. It was clear that the valve lift affected the flow, but there were some exception near the port blend upstream. It was also stated that the flow separation turns vital with valve lift. This was because of the losses increased with the increase in the valve lift 3. Xiangrong Li et al. researched about the lateral swirl combustion system (LSCS), which improved the soot emission, combustion and fuel efficiency in a direct injection diesel engine. Numerical simulation was implemented to further improve the LSCS performance and to analyse the effect of LSCS chamber geometries on combustion and emission characteristics under the condition of 2500 r/min and full load, exposing the relevant influence mechanisms. Accomplishment of chamber geometry optimization was completed which were based on the sensitivity analysis based on indicated power. The wall surface and the fuel spray jet has interplay in between, various fuels have different results. LSCS combustion performance had an effect on Split-flow geometry, where a dominant role was played by θ. Favourable flow guidance was created in the combustion chamber when θ was in 15–27^{o}range. After the geometrical optimization of the LSCS, fuel consumption decreased by 2.8 to 4.1 g/(kW.h) and soot emission decreased with the range between 69-75% under various engine speeds, as compared with the double swirl combustion system 13.

Federico Perini et al. studied the in-cylinder flow structure, in a swirl supported engine equipped with different piston geometries. The piston was having a conventional re-entrant bowl, first one had a cut-out valve and the second one had a cut-out valve and the third one had stepped-lip bowl. In an optical engine, particle image velocimetry was carried out to measure structure during intake and compression stroke and the swirl intensity. FRESCO code was used to build the optical diesel engine for computational model. Validation of the model was carried out in swirl-plane velocity field and multiple cycles were assessed for simulation overlapping. Study of flow topology was carried out which addressed the turbulence quantities and the bulk flow, swirl structure, operating parameters, squish flux and geometrics. It was observed that the conventional re-entrant bowls at intake had strong flow separation. Enclosed shape had better and strong squish flow, because of which neat TDC was stronger and 10% larger than SL bowl; Though it had more prominent global swirl than stepped-lip bowls, which was mainly because of lesser tilted swirl and stronger and more axisymmetric squish mechanism. Stepped-lip bowls had higher turbulence level and more in homogeneities. At the same time, it had faster turbulence at the TDC. With the valve cut-outs, velocity is large and the hit the piston surface which reduced the swirl ratio by 1.3% and turbulence increased by 4.6% 15. Guixin Wang et al. investigated a controllable intake swirl, its intake flow field in a diesel engine by using CFD. Two level characteristics were observed with the opening of the intake baffle with the variation of the intake swirl; though cut-off point of the baffle opening was 48^{0}. In the analysis of the baffle opening and valve lift, the flow coefficient was influenced and it was found that flow coefficient was more sensitive to valve lift. To calculate the controllable intake swirl and its characteristics in a diesel engine, mathematical model was used. A

combustion characteristic, which was influenced by swirl ratio was investigated. It was concluded that, in a diesel engine, the power performance had too much influence with swirl ratio. 5.79% increase in power performance was observed when the swirl ratio changed from 0.4 to 1.2, following which improvement in fuel consumption was observed 16. Alper Tolga Calik et al. studied MR-Process combustion chamber, which is different from the conventional combustion chamber in a diesel engine. Conventional CC has single swirl where as in case of MR-process it has twin swirl, helps in boosting better air fuel mixture. It is done by fuel spray vaporisation which is tangential to the piston wall. In the beginning of the study, a combustion chamber geometry of 2-valve was modified, to create twin swirl, known as Quasi MR-Process. The study summarised that, for twin swirl formation, a 4-valve engine was required. Designing an engine head with 4 valves (with two intake manifold), which would satisfy the ideal swirl condition in the combustion chamber, was a challenging task. Optimum values for swirl condition, air flow field and injection characteristics, experimental data were not available for MR-Process combustion chamber. Twin swirl initiation feasibility was investigated by numerical analysis of MR-Process combustion chamber. Twin swirls potential application was revealed in this study, utilizing closed cycle simulation. Existing swirl model (KIVA3V-R2) was altered to generate twin swirl, during the start of compression stroke. Initial swirl with different angular velocities and fuel spray injection directions was employed get the favourable air fuel mixture and to increase efficiency and lower the harmful gases. Results showed that MR-Process had potentiality to give better air fuel mixture and fuel efficiency, which would reduce the emission of harmful gases 18. Dan Moldovanu et al. optimise the combustion process in a compression ignited engine so that it could reduce the emission. Optimization was only possible by assuring a better air fuel mixture. Analysis of velocity and vertices formation inside the combustion chamber, in thirteen cases was carried out for numerical analysis using AVL FIRE Software. Software was used to create pure swirl motion, pure tumble motion, no air motion in in-cylinder and different combinations of swirl and tumble motion. Tumble and swirl motion cases were chosen combine 0%, 50%, 70% and 100%. At 100% tumble and 50% swirl motion, highest air motion (77.2 m/s at 736 degrees CA) was noted. While lowest velocity was noted when 100% swirl and 0% tumble, i.e., 71.07 m/s at 736 degrees, CA Various induced motion create vertices, which had influence on the air fuel mixture 19.

3. CAD MODEL

A 3D model of the cylinder, inlet port and valve was created with the help of Solidworks 2017. The model is shown in Figure 2. The dimension of the model is listed in Table 1. Figure 3 shows the imported CAD geometry in the Ansys software. During the geometry creation, 3 different post planes were created. The distance of the post planes from the reference planes were 30mm, 45mm and 60mm. The inlet plenum size was 100mm and the type of plenum was hemisphere, as shown in Figure 3. Three different types of valves were generated in Solid works, with different angles, 30^0,45^0 and 60^0.

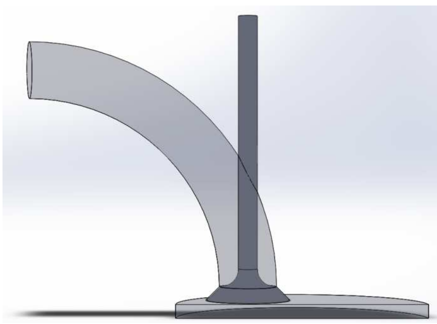


Figure 2: CAD File for Port Flow Simulation.

Table 1: 3D Model Parts Dimension

Sl. No.	Part Name	Value
1	Diameter of inlet port	25.4 mm
2	Diameter of outlet port	25.4 mm
3	Diameter of the cylinder	89.90 mm
4	Length of the cylinder	110 mm
5	Valve lift	6 mm
6	Valve angle	$30^0;45^0;60^0$

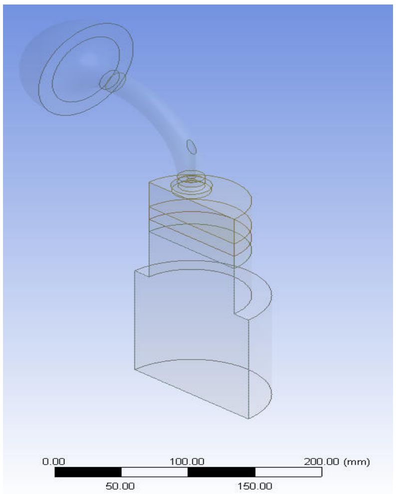


Figure 3: Imported CAD Geometry in Ansys.

4. MESHING

The 3D model was generated in Solidworks version 2018. To solve the problem, the geometry has to be decomposed in three stages. In the first stage, the geometry is decomposed in three stages and then, they are meshed. In the second step, with the use of the setup journal, the engine case is setup and in the final step, simulation is performed.

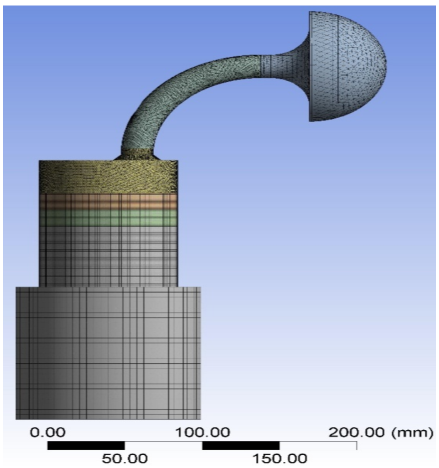


Figure 4: Meshed Geometry.

Most of the surface of the geometry was meshed with triangular element; the element size was 1.415 mm and the local min size was considered to be 0.315mm. In case of cylinder wall and the output plenum, the mapped mesh type was Hexa/Prism with 15mm minimum edge length. Number of elements used for generation of mesh was 1088684.

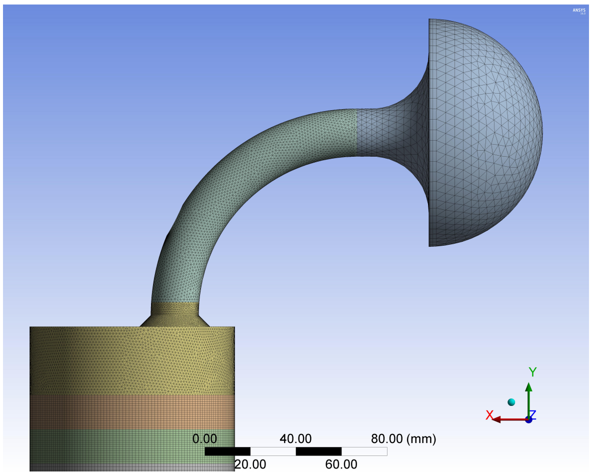


Figure 5: Meshing with Triangular Element.

5. MATHEMATICAL EQUATIONS

To solve a computational fluid dynamics problem, three equations are mainly used such as Energy Equation, Continuity Equation and Navier stroke Equation (momentum eq).

5.1 Energy Equation

This equation demonstrates that, per unit volume, the change in energy of the fluid moving through a control volume is equal to the summation of rate of heat transferred into the control volume, rate of work done by surface forces and work done because of gravity 3.

$$\frac{\partial}{\partial t}\left(\rho e+\frac{1}{2}\rho v^2\right)+\frac{\partial}{\partial x}\left(\rho ue+\frac{1}{2}\rho uv^2\right)+\frac{\partial}{\partial y}\left(\rho ve+\frac{1}{2}\rho vv^2\right)+\frac{\partial}{\partial x}\left(\rho we+\frac{1}{2}\rho wv^2\right)=k\left(\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}+\frac{\partial^2 T}{\partial xz^2}\right)-\left(u\frac{\partial p}{\partial x}+v\frac{\partial p}{\partial y}+w\frac{\partial p}{\partial z}\right)+\mu\left[u\frac{\partial^2 u}{\partial x^2}+\frac{\partial}{\partial x}\left(v\frac{\partial v}{\partial x}+w\frac{\partial w}{\partial x}\right)+v\frac{\partial^2 u}{\partial y^2}+\frac{\partial}{\partial y}\left(u\frac{\partial u}{\partial x}+w\frac{\partial w}{\partial y}\right)+w\frac{\partial^2 u}{\partial z^2}+\frac{\partial}{\partial z}\left(u\frac{\partial u}{\partial z}+v\frac{\partial v}{\partial z}\right)\right]+2\mu\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial u}{\partial y}\frac{\partial v}{\partial x}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial v}{\partial z}\frac{\partial w}{\partial y}+\frac{\partial^2 w}{\partial z^2}+\frac{\partial w}{\partial x}\frac{\partial u}{\partial z}\right]+\rho ug_x+\rho vg_y+\rho wg_z$$

5.2 Continuity Equation

A continuity equation expresses a conservation law by "Equating a net flux over a surface with a loss or gain of material within the surface". Continuity equations is shown below 3.

$$\int_{cs}\rho V dA+\frac{\partial}{\partial t}\int_{cv}\rho dA=0$$

This equation is for principle of mass conversation for a one dimensional, steady, with one inlet and outlet.

$$\nabla(\rho V)+\frac{\partial\rho}{\partial t}=0$$

Where, $\nabla=\frac{\partial}{\partial x}\hat{\imath}+\frac{\partial}{\partial x}\hat{\jmath}+\frac{\partial}{\partial x}\hat{k}$

$$\frac{\partial\rho}{\partial x}+\frac{\partial(\rho.u)}{\partial x}+\frac{\partial(\rho.v)}{\partial y}+\frac{\partial(\rho.w)}{\partial z}=0$$

5.3 Navier Stroke Equation (Momentum Eq)

The momentum equation states that sum of forces acting on a fluid element to its acceleration or rate of change of momentum. The Newton's second law of motion F = ma is the basis of the momentum equation **3**.

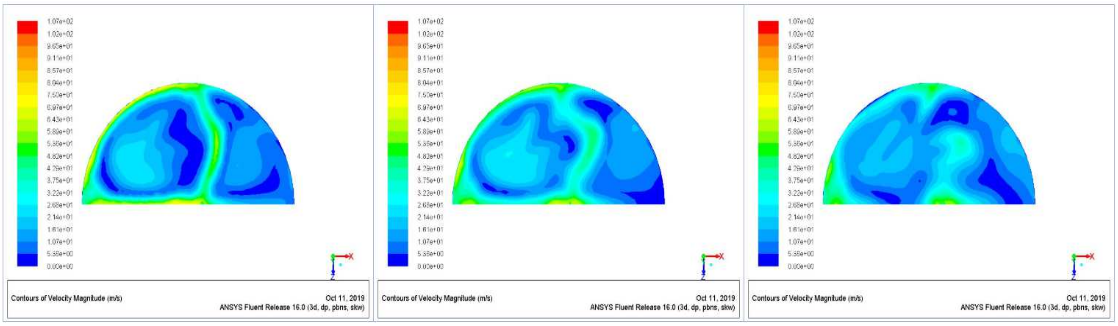
$$\rho\frac{\partial u}{\partial t}+\rho u\frac{\partial u}{\partial x}+\rho v\frac{\partial u}{\partial y}+\rho w\frac{\partial u}{\partial z}=\rho g_x-\frac{\partial p}{\partial x}+\mu\frac{\partial^2 u}{\partial x^2}+\mu\frac{\partial^2 u}{\partial y^2}+\mu\frac{\partial^2 u}{\partial z^2}$$

$$\rho\frac{\partial v}{\partial t}+\rho u\frac{\partial v}{\partial x}+\rho v\frac{\partial v}{\partial y}+\rho w\frac{\partial v}{\partial z}=\rho g_y-\frac{\partial p}{\partial y}+\mu\frac{\partial^2 v}{\partial x^2}+\mu\frac{\partial^2 v}{\partial y^2}+\mu\frac{\partial^2 v}{\partial z^2}$$

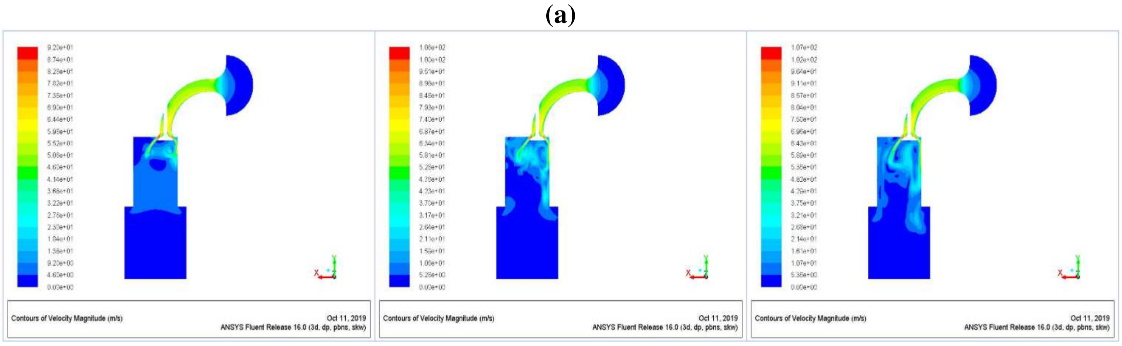
$$\rho\frac{\partial w}{\partial t}+\rho u\frac{\partial w}{\partial x}+\rho v\frac{\partial w}{\partial y}+\rho w\frac{\partial w}{\partial z}=\rho g_z-\frac{\partial p}{\partial z}+\mu\frac{\partial^2 w}{\partial x^2}+\mu\frac{\partial^2 w}{\partial y^2}+\mu\frac{\partial^2 w}{\partial z^2}$$

6. RESULTS AND DISCUSSIONS

The problem was solved with the help of using Ansys Fluent. The different valve angles (30^0; 45^0 & 60^0) were compared with each other. Below are the pictures of the air velocity inside the chamber. Figure 6Figure **8** show the velocity profile at different reference planes and timings. Figure 9–11 show the mass flow rate at different reference planes of different valves. Figure 12 shows the average mass flow rate.

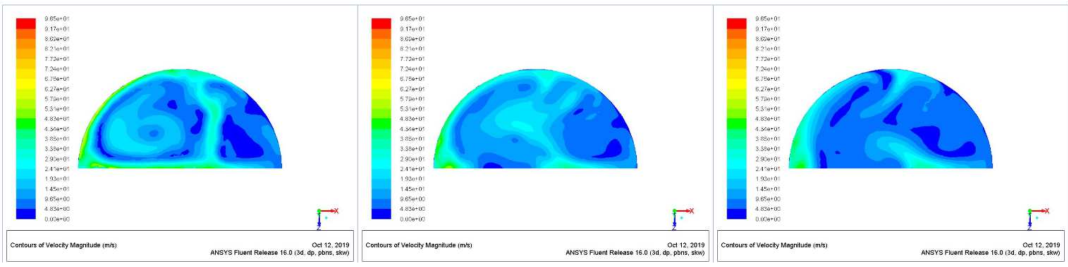


(a)

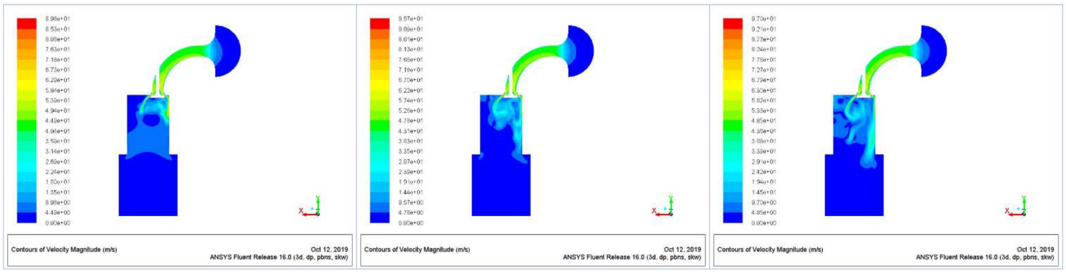


(b)

Figure 6: Velocity Profile at 45^0 Valve Geometry, (a) Velocity Profile at Different Reference Planes, (b) Velocity Profile at Different Time.

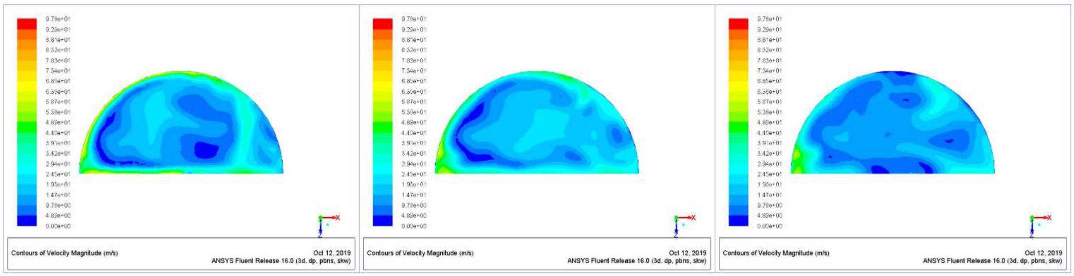


(a)

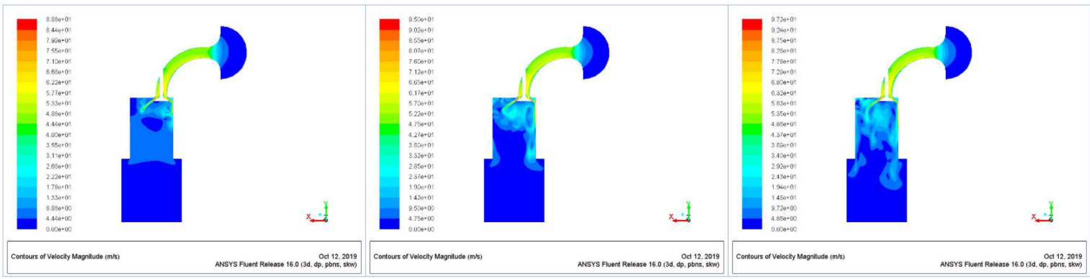


(b)

Figure 7: Velocity Profile at 60^0 Valve Geometry, (a) Velocity Profile at Different Reference Planes, (b) Velocity Profile at Different Time.

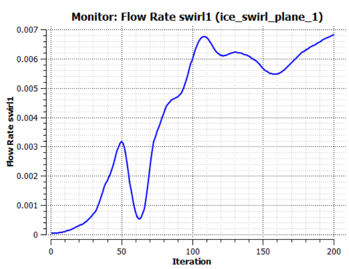


(a)

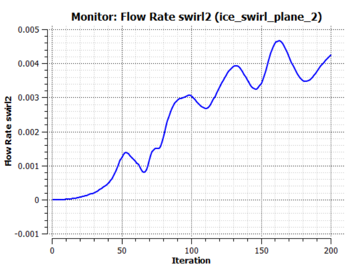


(b)

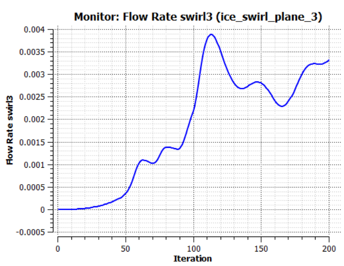
Figure 8: Velocity Profile at 30^0 Valve Geometry, (A) Velocity Profile at Different Reference Planes, (B) Velocity Profile at Different Time.



(a)

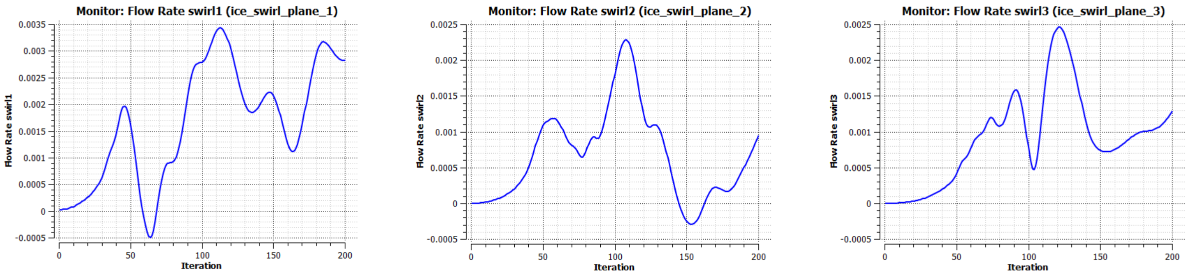


(b)



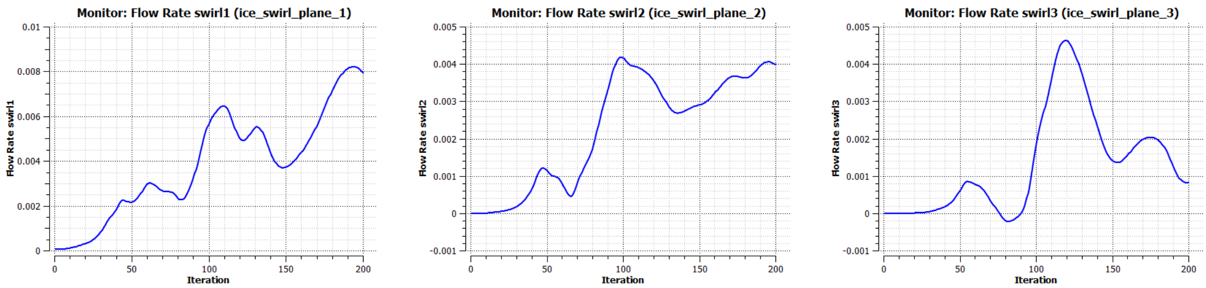
(c)

Figure 9: Velocity Profile at 45^0 Valve Geometry at 30mm, 45mm & 60mm Reference Plane Respectively.



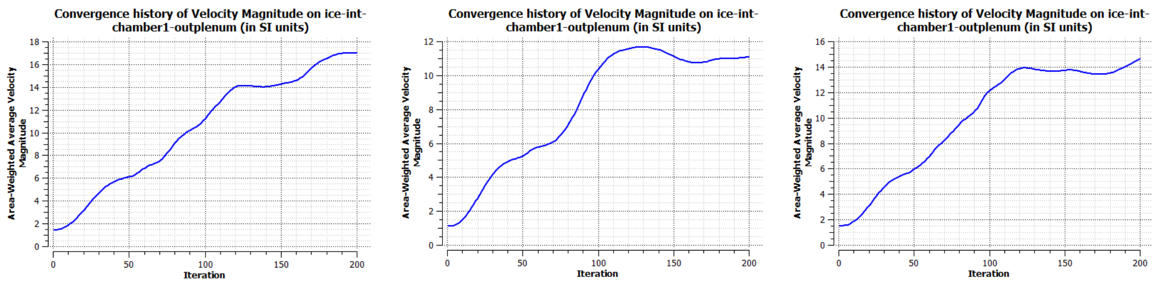
(a) (b) (c)

Figure 10: Velocity Profile at 60^0 Valve Geometry at 30mm, 45mm & 60mm Reference Plane Respectively.



(a) (b) (c)

Figure 11: Velocity Profile at 30^0 Valve Geometry at 30mm, 45mm & 60mm Reference Plane Respectively.



45^0 Valve Geometry 60^0 Valve Geometry 30^0 Valve Geometry

Figure 12: Area-weighted Average Velocity of Different Angle of Valve Geometry.

7. CONCLUSIONS

Air Fuel Mixture is very important for combustion to take place. Intake port plays a very critical function in the air flow inside the combustion chamber. The flow of air inside the combustion chamber helps in reaching better fuel efficiency and lower emission. This paper studied the valve design which could help in increasing the swirl and the mass flow rate in the combustion chamber. The study was concluded with the help of CFD IC engine module, Ansys. The valve angle was changed to 30^0, 45^0 & 60^0. Different CAD files were generated with the help of Solid works, the CAD files were imported into Ansys V16 in the IC Engine module. Port flow simulation was conducted on the generated files. Following conclusion was concluded from the analysis.

- When the port angle was set to 45^0, the swirl was limited to the upper region of the combustion chamber and the mass flow rate was higher
- In the second case, when the post angle was set to 60^0, the swirl in the chamber was better that 45^0 port angle, and the mass flow rate was less than the first condition

- In the third case, the post angle was set to 30^0. The swirl was less but the mass flow rate was higher.

Because of different angle of flow, the swirl and the mass flow rate changed. But from the above results, we could conclude that 45^0 port angle was a better option for swirl. Mass flow rate though 60^0 port angle should also be considered and experimental validation should be carried out in the future research.

8. ACKNOWLEDGEMENT

This work was supported by the National Research Foundation of Korea (NRF) grant funded by the Korea government (MSIT) (NRF-2019R1A2C1010557).

REFERENCES

1. *Haiqin Zhou, Xiangrong Li, Weihua Zhao, Fushui Liu. (2019). Effects of separated swirl combustion chamber geometries on the combustion and emission characteristics of DI diesel engines. Fuel 253. 488–500.*
2. *Mohamad Issa, Hussein Ibrahim, Richard Lepage, Adrian Ilinca. (2019). A Review and Comparison on Recent Optimization Methodologies for Diesel Engines and Diesel Power Generators. Journal of Power and Energy Engineering. 7. 31–56.*
3. *Himanth Kumar H Y, Jayashankar N. 2015. Port Flow Simulation of an Ic Engine. International Journal of Innovations in Engineering Research and Technology. 2. 9.*
4. *Arulprakasajothi Mahalingam, Yuvarajan Devarajan, Santhanakrishnan Radhakrishnan, Suresh Vellaiyan, Beemkumar Nagappan. 2018. Emissions analysis on mahua oil biodiesel and higher alcohol blends in diesel engine. Alexandria Engineering Journal. 57. 2627–2631.*
5. *A. Kalpakli Vester, Y. Nishio, P. H. Alfredsson. 2019. Investigating swirl and tumble using two prototype inlet port designs by means of multi-planar PIV. International Journal of Heat and Fluid Flow 75. 61–76.*
6. *Mohammad Abdullah Mohamad Johar, Mohd Farid Muhamad Said, Mohd Azman Abas, Zulkarnain Abdul Latiff, Mas Fawzi Ali, Nik Mohd Izual Nik Ibrahim. 2019. Flow Analysis of Intake Port Geometry of Spark Ignition Engine Using Simulation. Proceedings of the 3rd International Conference on Automotive Innovation Green Energy Vehicle. AIP Conf. Proc. 2059.*
7. *Thomas, J. O. S. E. P. H. (2014). Case study of error analysis of the usage of tense in English by year 1 engineering students from Tamil Medium Schools. IMPACT: International Journal of Research in Humanities, Arts and Literature, 2(3), 47–52.*
8. *David Rathnaraj Jebaman, Thathapudi Michael Narendra Kumar. 2008. Studies on Variable Swirl Intake System for Di Diesel Engine using Computational Fluid Dynamics. Thermal Science. 12. 25–32.*
9. *Soonseong Hong, Byungkeun Oh, Yongtae Kim, Wondae Jeon, Bongyong Jin. 2007. Optimization of Intake Manifold Design using DFSS (Design for Six Sigma). 14th Asia Pacific Automotive Engineering Conference Hollywood.*
10. *Rajasekaran, P., Vanangamudi, S., & Naveenchandran, P. Automation of hydraulic tyre wear testing machine using proportional control valve by interfacing programming logic circuit.*
11. *S. Bari, P. J. G. Johansen, A. J. T. Alherz. 2015. Simulation of improvements to in-cylinder mixing of biodiesel with air by incorporating guide vanes into the air intake system. Procedia Engineering 105. 480–487.*
12. *Haiqin Zhou, Xiangrong Li, Weihua Zhao, Fushui Liu. 2019. Effects of separated swirl combustion chamber geometries on the combustion and emission characteristics of DI diesel engines. Fuel 253. 488–500.*
13. *Xiangrong Li, Yanlin Chen, Liwang Su, Fushui Liu. 2018. Effects of lateral swirl combustion chamber geometries on the combustion and emission characteristics of DI diesel engines and a matching method for the combustion chamber geometry. Fuel 224. 644–660.*

14. *Arunachala, U. C., Rajat, A. C., Shah, D., & Sureka, U. (2019). Stability improvement in natural circulation loop using tesla valve–an experimental investigation. International Journal of Mechanical and Production Engineering Research and Development, 9(6), 13–24.*

15. *Federico Perini, Kan Zha, Stephen Busch, Eric Kurtz, Richard C Peterson, AlokWarey, Rolf D Reitz. 2018. Piston geometry effects in a light-duty, swirl-supported diesel engine: Flow structure characterization. International J of Engine Research .19(10). 1079–1098.*

16. *Guixin Wang, Wenbin Yu, Xiaobo Li, Yanpan Su, Rui Yang, Wentao Wu. 2019. Study on dynamic characteristics of intake system and combustion of controllable intake swirl diesel engine. Energy 180. 1008–1018.*

17. *Sakharkar, R. P., & Tasgaonkar, P. P. MPLS-traffic engineering load balance algorithm using uncommon path.*

18. *Alper Tolga Calik, Ozgur Oguz Taskiran, Rafig Mehdiyev. 2018. Numerical investigation of twin swirl application in diesel engine combustion. Fuel 224. 101–110.*

19. *Dan Moldovanu, Florin Mariaşiu, Norbert Bagameri. Influence of swirl and tumble motion inside the combustion chamber of a compression ignited engine on vertices formation. MATEC Web of Conferences 184, 01022.*

AUTHOR PROFILES

Adhirath Mandal has completed his bachelors in Mechanical & Automation Engineering from Guru Gobind Singh Indraprastha University, Delhi, India. After his graduation he has worked in a water RO Company for 1 year. He completed his M.Tech (masters) in production engineering from Delhi Technological University, Delhi, India. He has worked as Assistant Professor in engineering colleges in Delhi NCR. He has teaching experience of 3.5 year. He is pursuing his doctoral degree from Kongju National University, Cheonan, Korea in mechanical engineering. His area of interest is Biodiesel and electric vehicles. He has published papers in national and international journals. The papers are indexed in Scopus.

Professor HaengMuk Cho (Corresponding Author) is a renowned professor in South Korea. He is working as professor at Kongju National University in South Korea. He did Post Doctorate Fellowship from Department of Automotive Engineering of Loughborough University in UK and North Texas University in USA. He earned his PhD from Hanyang University, South Korea in 1997 and completed his M.D. in Environment Engineering from Yonsei University in S Korea. He is working on national and international projects in the field of Bio-fuels for IC engines and

renewable energy. Professor HaengMuk Cho is working in the field of Renewable Energy and Biofuels. He is also a member of many societies, like Korea Society of Automotive Engineers, Korean Society of Energy (Korea), Korea Society of Marine Engineering (Korea) & Korea Society of Environment Engineer (Korea). He has published many national and internal research papers. Most of his papers are indexed in Scopus and SCI.